DCIEM CR-2000-002

Modelling of Thermal, Mechanical and Fluid Dynamics of an Open-Circuit Underwater Demand Breathing Regulator

Framework Design and Development

by
Z.Wozniak

Coupled Systems Ltd.
7033 Bonnie Street
Niagara Falls, Ontario, L2G 7N1
Phone/Fax:(905)354-7186

PWGSC Contract No. W7711-9-7530/001/TOR

on behalf of
DEPARTMENT OF NATIONAL DEFENCE

as represented by
Defence and Civil Institute of Environmental Medicine
1133 Sheppard Avenue West
North York, Ontario, Canada
M3M 3B9

DCIEM Scientific Authority:
David Eaton, M.SC., P.Eng.
Phone:(416)635-2086

November 1999

©HER MAJESTY THE QUEEN IN RIGHT OF CANADA (1999)
as represented by the Minister of National Defence
©SA MAJESTE LA REINE ENDROIT DU CANADA (1999)
Défence Nationale Canada

Contents

I User's Manual 1

1 Program Organization 2
- 1.1 Installing SCUBA . . . 2
- 1.2 Create AutoCAD Drawing . . . 2
- 1.3 FEMAP Mesh Generation . . . 3
- 1.4 Finite Element Solver . . . 3
- 1.5 Data Visualization . . . 4

2 Input Data Format 5
- 2.1 Nodal Data . . . 5
- 2.2 Material Properties . . . 6
- 2.3 Element Data . . . 7
- 2.4 Boundary Conditions . . . 8
- 2.5 Load Conditions . . . 8

II Examples Manual 13

3 Introduction 14
- 3.1 Solid Modelling . . . 14

III Theoretical Manual 17

4 Model Formulation 18
- 4.1 Variational Principles . . . 18
- 4.2 Boundary Conditions . . . 19
- 4.3 Constitutive Relations . . . 19
  - 4.3.1 Elastic Solid . . . 19
  - 4.3.2 Fluid . . . 20

5 Finite Element Formulation 21
- 5.1 Element Matrices . . . 21
- 5.2 Surface Loads . . . 22
- 5.3 Element Loads . . . 22
  - 5.3.1 Thermal Stresses . . . 22
  - 5.3.2 Gravity Body Force . . . 22

List of Figures

3.1 Pipe Segment (8-node brick) 15
3.2 Pipe Deformation 15

List of Tables

2.1 Element Shapes 10
2.1 Element Shapes, Cont. 11
2.2 Element Face Definition 12

3.1 Displacement Summary $\times 10^{-9}$ (m) 16

Abstract

SCUBA is a finite element code developed for Defence and Civil Institute of Environmental Medicine under research contracts with Canadian industry from 1998 to the present. Its purpose is to provide a computer aided design facility for modelling of thermal, mechanical and fluid dynamics of an open-circuit underwater demand breathing regulators. This document comprises a set of three parts that provides information on SCUBA:

1. *User's Manual* - outlines the input and output data specification, describes the element library and storage allocations.
2. *Examples Manual* - provides a number of sample problems to aid users in model development and output interpretation. It demonstrates virtually all elements and features of SCUBA package.
3. *Theoretical Manual* - provides background material on the theoretical basis of the software, including details on many facets of model construction and operation. The program solution capabilities, the numerical techniques used and logical construction of the program are discussed.

The SCUBA package is written in C++, and will run on any IBM 386 PC compatible platforms without further modifications. SCUBA software is supported by Coupled Systems Ltd.

Part I

User's Manual

Chapter 1

Program Organization

The complete solution process of solving is divided into four distinct stages:

1. **Create AutoCAD drawing.**
2. **FEMAP mesh generation.**
3. **Running the solver.**
4. **Data visualization.**

Each stage together with the installation guide is briefly described in the following sections.

1.1 Installing SCUBA

The SCUBA package is an assemblage of the ready to use off the shelf software packages such as AutoCAD and FEMAP, and the parts developed for SCUBA such as the solver and the graphical user interface (GUI) which manages the package. In order to function properly, each of these parts has to be installed in the appropriate directory on the computer hard disk. The SCUBA's default main directory is `C:\SCUBA`. Within this main directory, a number of subdirectories should be created, each for every SCUBA's part as follows:

1. `C:\SCUBA\AUTOCAD` - AutoCAD executable file,
2. `C:\SCUBA\FEMAP` - FEMAP executables,
3. `C:\SCUBA\NOTEPAD` - NOTEPAD Window text editor,
4. `C:\SCUBA\RUNS` - SCUBA executables together with project input/output data.

SCUBA can be executed from within the `C:\SCUBA\RUNS` subdirectory. If the user wishes to change these default settings, the code has to be modified accordingly and recompiled. It is a very simple operation and can be done on site.

1.2 Create AutoCAD Drawing

Starting point in the regulator design and analysis process is a generation of the assembly drawings together with production drawings for manufacturing components. The drawing process in SCUBA is initiated by selecting the *AutoCAD* option from *Pre/Post* menu. It opens the AutoCAD environment where the user can create drawings in the similar way as it would be done on the paper. When this task

is completed, the regulator geometry is stored in a digital form (*.DXF file) using "Save As.." option within the AutoCAD environment. These drawings stored digitally provide input to the next task which is the mesh generation.

1.3 FEMAP Mesh Generation

The most tedious and time consuming stage in the input data preparation for the finite element analysis is the mesh generation. The meshing is a form of discretization where the solid parts of the regulator are partitioned into a large number of much smaller elements. Each element, in turn, is defined by a set of the points located at the element corners. The element assembly defines the regulator geometry. Meshing in SCUBA is initiated by selecting the *FEMAP* option from *Pre/Post* menu. FEMAP is a specialized software package which provides aid to the analyst in generating the finite element mesh. From within the FEMAP environment the user may import the regulator geometry stored in DXF files generated previously by AutoCAD. If drawings are not available, the mesh can also be generated by entering the regulator geometry directly. A very efficient automesh feature available in FEMAP may speed up the meshing process. It should be noted that the drawing provides only the wire frame geometry which has to be processed further by creating solids, shells, beams, and other structural elements from these basic geometry data. The same package is used later also for the visualization of the results.

1.4 Finite Element Solver

The solving sequence is executed in four distinct phases:

1. *Finite element mesh and element data input.* In this phase the control information and the nodal point input data are read and generated by the program. The equation numbers for the active degrees of freedom at each nodal point are established. The initial conditions are read. The element data are read and generated, the element connectivity arrays are calculated and all element information is stored on disk.

2. *Assemblage of constant structure matrices.* Before the solution to the problem is carried out, the linear structure stiffness, mass, and damping matrices are assembled and stored on disk.

3. *Load vector calculations.* The externally applied load vectors for each time (load) step are calculated and stored on tape.

4. *Step-by-Step Solution.* During this phase the solution is obtained at all time points. In addition to the displacement, the nodal temperatures and fluid velocities together with fluid nodal pressure are computed.

It needs be noted that these basic steps are independent of the element type used and are the same for either a static or dynamic analysis.

The analysis part in SCUBA is initiated by selecting a task from the *Runs* main menu. From the drop down selection list the user may pick one of the following tasks:

1. Thermal analysis,
2. Fluid flow,
3. Thermal and fluid flow analysis,
4. Chatter,
5. Static.

At this stage of development only the *Static* option is available. Upon selection, the user is prompted to enter the project name which is used in the program as a keyword for creating a number of output files which can be examined later either by inspection or by using FEMAP to generate the deformation plots.

1.5 Data Visualization

Visualization of the results is accomplished using FEMAP package for post-processing. The results are imported to FEMAP using *Comma Separated Table* option which reads the columns of nodal or elemental data generated previously by one of the executed tasks. The *Static* run generates automatically the file in the CSV format compatible with FEMAP so the user can visualize the deformation immediately after the run is completed.

Chapter 2

Input Data Format

Data is input to solver in a file *project*.dat that is divided into *cards*. A *card* is a record of up to 132 characters, the name being derived from the now historical procedure of using punched cards for data input. The input data records for solver are in free format, which makes the data preparation more user friendly.

This chapter describes the format and data value descriptions card-by-card: First is a table giving the name of the variable, its size if it is an array variable, and the order in which it is placed in the 132-character record; next, a brief description of the variable is given with allowable values stated when applicable; finally some explanatory comments, warnings, and suggestions are given. The data are entered using **free format**, thus all entries have to be defined explicitely. **No blank entry** is allowed in a free format input.

The input data is divided into five major sections:

1. Nodal coordinates,
2. Material property data sets,
3. Elements data,
4. Boundary conditions,
5. Load definition.

2.1 Nodal Data

The following card controls the memory allocation, and other features available in SCUBA.

```
nPoin nDim
```

where
nPoin number of nodes
nDim spatial dimension (=2 for 2D, =3 for 3D)

Nodal Coordinates

The nodal coordinates are entered in this subsection. One card is required for each node:

```
node xyzG(node,1) xyzG(node,2) xyzG(node,3)
```

where

node	node sequential number
xyzG(node,1)	node X-coordinate
xyzG(node,2)	node Y-coordinate
xyzG(node,3)	node Z-coordinate

NOTES: Number and type of degrees of freedom associated with the node (displacements (u, v, w) are determined automatically within the program.

2.2 Material Properties

Element material properties are entered in 2 groups of cards defining

- Material Density and Thermal Expansion Coefficients,
- Elastic Constitutive Matrix,

These are repeated for each individual material.

```
nMat
```

where nMat is a number of material property sets.

```
mType
```

where mType is a material type number. Currently only mType=1 (elastic solid) is allowed. In the next stages of development the fluid material and acoustic material type sets will be introduced.

```
dns alf1 alf2 alf3
```

where

dns	density (kg/m3)
ALF(1,2,3)	coefficients of thermal expansion $/°C$ in local X-, Y-, and Z-direction, respectively.

Elastic Constitutive Matrix

Six cards are required for each material type:

```
CST(1,1) CST(1,2) CST(1,3) CST(1,4) CST(1,5) CST(1,6)
CST(2,1) CST(2,2) CST(2,3) CST(2,4) CST(2,5) CST(2,6)
CST(3,1) CST(3,2) CST(3,3) CST(3,4) CST(3,5) CST(3,6)
CST(4,1) CST(4,2) CST(4,3) CST(4,4) CST(4,5) CST(4,6)
CST(5,1) CST(5,2) CST(5,3) CST(5,4) CST(5,5) CST(5,6)
CST(6,1) CST(6,2) CST(6,3) CST(6,4) CST(6,5) CST(6,6)
```

where

CST(I,J)	Matrix of elastic stiffness moduli, c_{ij}, in Pascals.

NOTES:

1. Materials that have orthotropic or higher symmetry may be used (nine independent moduli). The matrix c_{ij} is always symmetric (i. e., $c_{ij} = c_{ji}$).

2. The material axes should be aligned with the axes. They can be rotated in 3D by arbitrary angles about the X-,Y-, and Z-global axis.

2.3 Element Data

Element specifications are entered in this section. One card is needed for each element.

```
iel ieltyp imtyp inc(1..nnod)
```

where

iel	element sequential number
ieltyp	element type. The element types are as listed in Table 2.1
imtyp	element material number; must correspond to one of those in the materials list.
inc(i)	element connectivity (between 2 and 20 node numbers). Ordering must correspond to the specifications in the element shown in Table 2.1.

Element Types

All elements in the model are constructed on the bases of volume, gap, surface, and line shapes. The base volume shapes are brick, wedge and tetrahedron; the base gaps are brick and wedge; the base surfaces are quadrilateral and triangular; and there is also a line shape. All elements were developed from the published shape functions [1, 6].
Upon these basic shapes are constructed two mutually exclusive sets of elements. Half are linear type elements with straight sides and nodes only at the corners. The other half are the corresponding parabolic elements, with added midside nodes. The parabolic elements may all have curved edges. An operational restraint that needs to be obeyed, however, is that each midside node be placed within 1/6 of the end to end length of the geometric midedge location, to preserve integration accuracy and solution integrity. This is commonly called the "middle third" rule.
Element formulations have been created in a mechanical engineering standard x-y-z coordinate system. Each of the basic element shapes are shown in Table 2.1.

2.4 Boundary Conditions

Boundary conditions are entered in this section. They specify the nodal fixities together with nodal concentrated loads. Distributed loads are described in the following. One card for each node where any of the fix > 1 is required:

```
node dof fix val
```

where

node node number

dof degree of freedom number. The sequence of DOF numbers shown in table below follows the general rule that the displacement degrees of freedom come first, followed by rotational degrees of freedom.

	dof
Displacement X	1
Displacement Y	2
Displacement Z	3
Rotation X	4
Rotation Y	5
Rotation Z	6

fix fixity or load codes specified for each DOF at a node
=0 free, no fixity, no load, no additional lumped mass
=1 DOF fixed to zero
=2 force
=3 nonzero fixity

val specify the values of fixity or load for each individual DOF.

2.5 Load Conditions

In modelling a complex coupled system such as the demand breathing regulator, the load conditions apply across the variables spectrum simultaneously. Currently the following load conditions are available:

1. Concentrated load,
2. Pressure Load.

One card is required to enter the number of load data sets:

```
nLoa
```

Immediately following are the cards defining the load data sets.

Concentrated Load

The concentrated load may be applied to the node. One card for each concentrated load plus a single card to indicate a number of loads are required:

```
nconc
```

```
node dof val
```

where

nconc	number of concentrated loads in this data set,
node	node number,
dof	degree of freedom (1-6),
val	force.

Static Pressure

The load component resulting from static pressure acting on the element surface is defined using an element number and element face numbers. One card for each element face plus the card which indicate how many pressure loads are required:

```
nPress
```

```
iel iface val
```

where

nPress	number of pressure loaded element faces,
iel	element number,
iface	element face number as shown in Table 2.2,
val	pressure.

This method of describing the pressure load can be applied to solid or shell elements only.

Table 2.1: Element Shapes

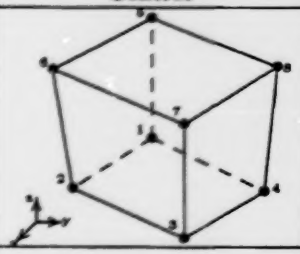
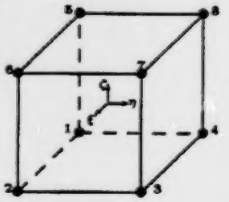
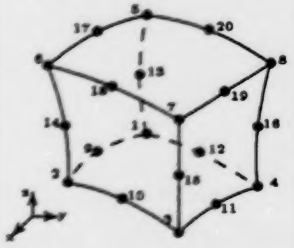
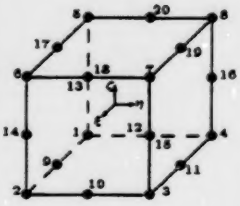
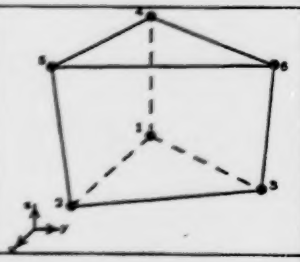
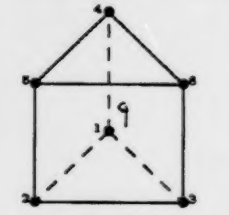
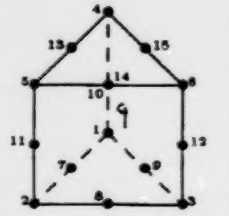
Name	General	Parent	Shape Functions
Brick Linear			$N_i = \frac{1}{8}(1+\xi_o)(1+\eta_o)(1+\zeta_o)$
Brick Parabolic			For corner nodes: $N_i = \frac{1}{8}(1+\xi_o)(1+\eta_o)(1+\zeta_o) \times (\xi_o+\eta_o+\zeta_o-2), i = 1,...,8$ For midside nodes: $N_i = \frac{1}{4}(1-\xi^2)(1+\eta_o)(1+\zeta_o)$ $i = 9, 11, 17, 19$ $N_i = \frac{1}{4}(1-\xi_o)(1-\eta^2)(1+\zeta_o)$ $i = 10, 12, 18, 20$ $N_i = \frac{1}{4}(1-\xi_o)(1+\eta_o)(1-\zeta^2)$ $i = 13, 14, 15, 16$
Wedge Linear			$N_i = \frac{1}{2}L_i(1+\zeta_o),$
Wedge Parabolic			For corner nodes: $N_i = \frac{1}{2}L_i(2L_i-1)(1+\zeta_o) - \frac{1}{2}L_i(1-\zeta^2), i = 1,...,6$ For midside nodes: $N_i = 2L_1L_2(1+\zeta_o), i = 7, 13$ $N_i = 2L_2L_3(1+\zeta_o), i = 8, 14$ $N_i = 2L_3L_1(1+\zeta_o), i = 9, 15$ $N_{10} = L_1(1+\zeta_o)$ $N_{11} = L_2(1+\zeta_o)$ $N_{12} = L_3(1+\zeta_o)$

Table 2.1: Element Shapes, Cont.

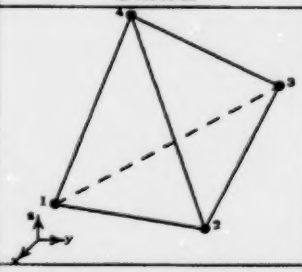
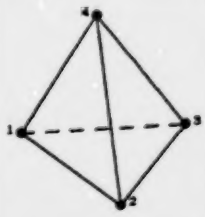
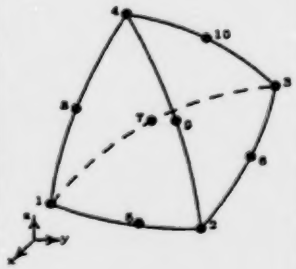
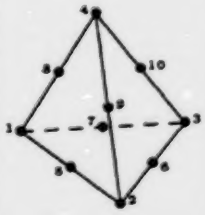
Name	General	Parent	Shape Functions
Tetrahedral Linear			$N_i = L_i$
Tetrahedral Parabolic			For corner nodes: $N_i = L_i(2L_i - 1); \quad i = 1,2,3,4$ For midside nodes: $N_5 = 4L_1L_2$ $N_6 = 4L_2L_3$ $N_7 = 4L_1L_3$ $N_8 = 4L_1L_4$ $N_9 = 4L_2L_4$ $N_{10} = 4L_3L_4$

Table 2.2: Element Face Definition

Face Number	Corner Nodes			
	i	j	k	l
Brick				
1	1	4	3	2
2	5	6	7	8
3	1	2	6	5
4	2	3	7	6
5	3	4	8	7
6	4	1	5	8
Wedge				
1	3	2	1	
2	6	4	5	
3	1	2	5	4
4	2	3	6	5
5	3	1	4	6
Tetrahedron				
1	3	2	1	
2	1	2	4	
3	2	3	4	
4	3	1	4	
Triangular Shell				
1	3	2	1	
2	1	2	3	
Quadrilateral Shell				
1	4	3	2	1
2	1	2	3	4

Part II

Examples Manual

Chapter 3

Introduction

The purposes of this Examples Manual are:

- to aid SCUBA users in model development, and data input and output interpretation,
- to provide comprehensive new user training,
- to exercise and demonstrate all elements and features of SCUBA package,
- to provide a set of problems to validate proper operation of the program after modifications are made,
- to help in establishing timing benchmarks for reference after modifications are made.

The manual contains many solved examples for SCUBA that test the code on the type of problems that users are likely to solve. Many problems are quite difficult and test a combination of capabilities in the code. In addition, this Examples Manual contains a number of examples that are intended as elementary verification of the finite element integrity. The input data for SCUBA have been designed to make it easy as possible to define the analysis to the computer.

3.1 Solid Modelling

The first test on the solid elements, Type 1, is a simple pressure load test to validate element stiffness representation and stress distribution. The geometry was a quarter of a thick aluminum pipe segment, with three adjoining faces constrained, subjected to an internal pressure load p = 1 MPa.

Material Properties

The aluminum material properties are as follows:

Density	ρ	2700 kg/m^3
Young's modulus	E	$7{\times}10^{10}$ Pa
Poisson's ratio	ν	0.3

Problem Dimensions

The dimensions of the pipe segment are as follows:

Internal radius	0.9 m
External radius	1.1 m
Height	1.0 m

Finite Element Mesh

The finite element model consisting of 192 8-node elements and 351 nodes is shown in Figure 3.1:

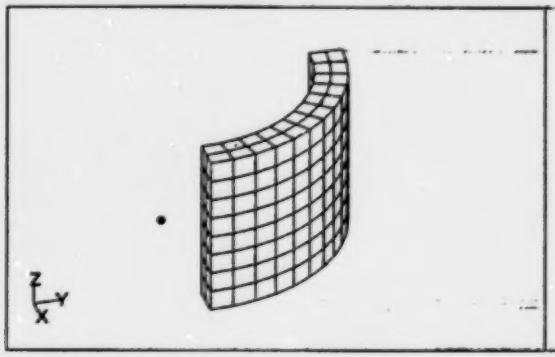


Figure 3.1: Pipe Segment (8-node brick)

Deformation

The exact solution to the internally loaded pipe segment is:

$$u_r = \frac{p_i}{E} \frac{r^2 r_i^2 (1-\nu) + r_i^2 r_o^2 (1+\nu)}{r(r_o^2 - r_i^2)} \tag{3.1}$$

where u_r is the radial deformation, p is the pressure, r is a radius, and i, o indicate internal or external quantity, respectively.

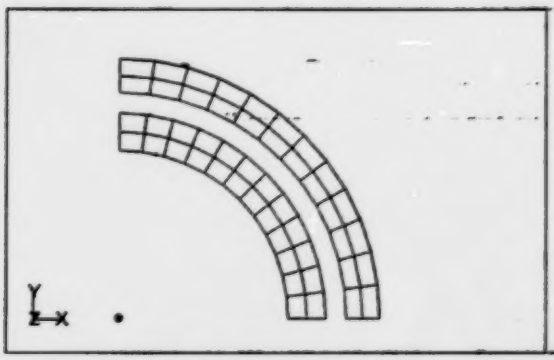


Figure 3.2: Pipe Deformation

Computed deformations were compared with theoretical results in Table (3.1) below:
The results shown in Table (3.1) are in a very good agreement with theoretical results. Because of fully symmetric orientation of elements with respect to the model symmetry plane, all results produced fully symmetric responses, with identical displacements in the three directions. This was taken to indicate proper functionality. These elements were performing properly, according to their ability.

Table 3.1: Displacement Summary $\times 10^{-9}$ (m)

Results	Wall Inside	Error %	Wall Outside	Error %	Number of Nodes	Number of Elements
Theory	**68.78**	-	**63.64**	-	-	-
8-node brick	68.68	0.14	63.57	0.11	351	192

Part III

Theoretical Manual

Chapter 4

Model Formulation

The essential components of the finite element model of the underwater demand breathing regulator are the model variables (degrees of freedom) related by equations of motion and the boundary conditions. These in turn are derived from the variational principles and the material constitutive relations. Following sections provide a brief description of the model components.

4.1 Variational Principles

The set of dynamic equations of motion of the structure can be derived from the Lagrange equations [3]

$$\frac{d}{dt}\frac{\partial \mathcal{L}}{\partial \dot{q}_n} - \frac{\partial \mathcal{L}}{\partial q_n} - \frac{\partial \mathcal{R}}{\partial \dot{q}_n} = \mathcal{F}_n \tag{4.1}$$

In the solid body, the generalized coordinates q_n are the components of the elastic displacements u_1, u_2, u_3.
where $\mathcal{L} = \mathcal{T} - \mathcal{U}$ is called the Lagrangian function, $\mathcal{T}$ is the kinetic energy of the body, $\mathcal{U}$ is the body potential energy, $\mathcal{R}$ is the dissipation function.
Mechanical variables are related through a constitutive relation for an elastic material given by Equation 4.2.

$$\sigma_i = c_{ij}\varepsilon_j \tag{4.2}$$

where σ is the mechanical stress and ε) is the elastic strain.
For a single element e, the kinetic energy $\mathcal{K}_e$ can be expressed as

$$\mathcal{K}_e = \frac{1}{2}\int_{V_e} (\rho \dot{\mathbf{u}}^T \dot{\mathbf{u}}) dV_e. \tag{4.3}$$

and potential energy $\mathcal{U}_e$ is composed of elastic energy

$$\mathcal{U}_e = \frac{1}{2}\int_{V_e} (\sigma^T \varepsilon dV_e. \tag{4.4}$$

where V_e is the volume of the element, ρ is the material density, $\mathbf{u}$ is the vector of velocities of element e, and a superscript T means transposed quantity.

The generalized forces $\mathcal{F}$ that contribute to Lagrange equations (4.1) can be obtained from

$$\mathcal{F} = \frac{\partial \mathcal{W}}{\partial \mathbf{u}} \tag{4.5}$$

where the work $\mathcal{W}$ of external loads is

$$\mathcal{W} = \int_V \mathbf{u}^T \bar{\mathbf{f}} dV + \int_{S_2} \mathbf{u}^T \bar{\mathbf{t}} dS_2 + \sum \mathbf{u}^T \bar{\mathbf{F}} \tag{4.6}$$

is defined by the loads
- $\bar{\mathbf{f}}$ specified body force vector,
- $\bar{\mathbf{t}}$ surface traction,
- $\bar{\mathbf{F}}$ mechanical point force load

The total body surface is $S = S_1 + S_2$ with portion S_1 where prescribed displacements are zero, and portion S_2 where the surface tractions are applied.
Assuming that the dissipative forces are proportional to the relative velocities, the dissipation energy of the element e can be expressed as

$$\mathcal{R}_e = \frac{1}{2} \int_{V_e} (\mu \dot{\mathbf{u}}^T \dot{\mathbf{u}}) dV_e. \tag{4.7}$$

where μ represents the body damping coefficient.

4.2 Boundary Conditions

Solution of equations of motion can be determined subject to the boundary conditions. In the most general case the following boundary conditions can be prescribed on the body surface

$$\sigma \mathbf{n} = \mathbf{t}_s \tag{4.8}$$
$$\mathbf{u} = \mathbf{u}_s \tag{4.9}$$

where $\mathbf{t}_s$ represents the surface traction, $\mathbf{u}_s$ is the prescribed displacement.

4.3 Constitutive Relations

Two material types are considered in the regulator modelling and analysis

- elastic solid, and
- fluid

4.3.1 Elastic Solid

For linear isotropic solid, the elasticity tensor is completely defined by only two constants [1]; Young's modulus E and the Poisson's ratio ν

$$c_{ij} = \frac{E}{(1+\nu)(1-2\nu)} \begin{pmatrix} 1-\nu & \nu & \nu & 0 & 0 & 0 \\ \nu & 1-\nu & \nu & 0 & 0 & 0 \\ \nu & \nu & 1-\nu & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1-2\nu}{2} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1-2\nu}{2} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1-2\nu}{2} \end{pmatrix} \tag{4.10}$$

The constitutive matrices, c_{ij} relates stresses to strains (Hooke's Law) by

$$\sigma_i = c_{ij}\varepsilon_j. \tag{4.11}$$

and the strains are in turn related to the elastic displacements u

$$\varepsilon = \frac{1}{2}(\nabla \mathbf{u} + \nabla \mathbf{u}^T). \tag{4.12}$$

Initial thermal strains may be specified as $\varepsilon^o_{ii} = \alpha_{ii}\Delta T$, where α is a vector of thermal expansion coefficients, and ΔT is the temperature increase above free stress reference temperature.
Gravity body loads may be specified. Inertia is specified by the density, and both mass and stiffness distributed damping may be included in the element formulation.

4.3.2 Fluid

The fluid elastic material properties are defined using the fluid density ρ and the fluid bulk modulus K_b is. The speed of sound in the fluid C_s can be computed internally using the formula

$$C_s = \sqrt{K_b/\rho} \tag{4.13}$$

It should be noted that in SCUBA acoustic fluid is not implemented yiet but in the future versions the acoustic fluid will be modelled using the fluid velocity potential formulation.

Chapter 5

Finite Element Formulation

The discretization of the variational principles by the finite element method results in the equations of motion for the solid body similar to those given by Zienkiewicz [1]

$$\mathbf{M}\ddot{\mathbf{u}} + \mathbf{C}\dot{\mathbf{u}} + \mathbf{K}\mathbf{u} = \mathbf{F}_B + \mathbf{F}_S + \mathbf{F}_P \tag{5.1}$$

where the global matrices **M**, **C**, and **K** represent the mass matrix, damping matrix, and stiffness matrix, respectively. The vectors $\mathbf{F}_B, \mathbf{F}_S, \mathbf{F}_P$ represent the body force, surface force, and concentrated load vector, respectively.

5.1 Element Matrices

The global matrices are assembled from the element matrices shown in the following formulas:

$$\begin{aligned}
\mathbf{M}^e &= \int_{V_e} \rho \mathbf{N}^T \mathbf{N} dV_e \\
\mathbf{C}^e &= \int_{V_e} \mu \mathbf{N}^T \mathbf{N} dV_e \\
\mathbf{K}^e &= \int_{V_e} \mathbf{B}^T \mathbf{c} \mathbf{B} dV_e
\end{aligned} \tag{5.2}$$

where V_e is the volume of the element e.
A general form of the strain-displacement relation is

$$\mathbf{S} = \mathbf{B}\mathbf{u} \tag{5.3}$$

where the matrix **B** for three-dimensional geometry is given by

$$\mathbf{B} = \begin{bmatrix}
\frac{\partial N_i}{\partial x} & 0 & 0 \\
0 & \frac{\partial N_i}{\partial y} & 0 \\
0 & 0 & \frac{\partial N_i}{\partial z} \\
\frac{\partial N_i}{\partial y} & \frac{\partial N_i}{\partial x} & 0 \\
0 & \frac{\partial N_i}{\partial z} & \frac{\partial N_i}{\partial y} \\
\frac{\partial N_i}{\partial z} & 0 & \frac{\partial N_i}{\partial x}
\end{bmatrix} \tag{5.4}$$

5.2 Surface Loads

The global surface load vector $\mathbf{F}_S$ comes from collecting the element surface load vectors $\mathbf{F}_S^e$

$$\mathbf{F}_S^e = \int_{S_e} \mathbf{B}^T \mathbf{n} p dS_e \tag{5.5}$$

where p is a pressure load distributed over the element surface S^e and $\mathbf{n}$ is the unit vector normal to the element surface.

5.3 Element Loads

In addition to the point loads and surface distributed loads, SCUBA allows specification of the volume distributed loads such as thermal and gravity loads.

5.3.1 Thermal Stresses

Initial thermal strains are allowed in addition to nodal and distributed pressure loads for all structural materials. The thermal load vector can be computed using

$$\mathbf{F}_T^e = \int_{V_e} \mathbf{B}^T \mathbf{c} \mathbf{S}_o dV_e \tag{5.6}$$

where $\mathbf{S}_o = \alpha \Delta T$ is the thermal strain defined by a vector of thermal expansion coefficients $\alpha = \{\alpha_{11}, \alpha_{22}, \alpha_{33}\}$, and the body temperature increase ΔT above the stress free body reference temperature.

5.3.2 Gravity Body Force

Body forces, which are distributed over the volume of elements, must be converted to equivalent nodal forces. They will then form part or all of the load vector. The gravity load portion of the load vector is computed using

$$\mathbf{F}_G^e = \int_{V_e} \rho g \mathbf{N}^T dV_e \tag{5.7}$$

where g is the gravity acceleration.

Bibliography

[1] *The Finite Element Method, O. C. Zienkiewicz and R.L. Taylor*, Fourth Edition, McGraw-Hill Book Company (UK) Limited, London, 1991.

[2] *Finite Element Procedures, K.J. Bathe*, Prentice-Hall, Inc., 1996.

[3] Chi Chin, L., Varadan, V.V., Varadan, V.K.*Hybrid Finite Element Formulation,*

[4] *Finite Element Handbook*, H. Kardestuncer, McGraw-Hill, 1987.

[5] Irons, B., Ahmad, S.,*Techniques of Finite Elements,* Ellis Horwood, Wiley, Chichester, 1980

[6] *The Finite Element Method in Engineering, S. S. Rao*, Pergamon Press,1982.

[7] Press, W.H, at al,*Numerical Recipes in C,* Cambridge University Press (1988).

[8] *ANSYS User Manual - Theory,* Vol. IV, 1994

[9] Segerlind, L.J.,*Applied Finite Element Analysis,* John Wiley & Sons, 1976

[10] Korn, G.A.,Korn, T.M.,*Mathematical Handbook for Scientists and Engineers,* McGraw-Hill, 1961